N88-15791

S1-34
118764
24P.

INLETS, DUCTS, AND NOZZLES

John M. Abbott,
Bernhard H. Anderson,
and
Edward J. Rice

ABSTRACT

The internal fluid mechanics research program in inlets, ducts, and nozzles is described. The program consists of a balanced effort between the development of computational tools (both parabolized Navier-Stokes and full Navier-Stokes) and the conduct of experimental research. The experiments are designed to better understand the fluid flow physics, to develop new or improved flow models, and to provide benchmark quality data sets for validation of the computational methods.

The inlet, duct, and nozzle research program is described according to three major classifications of flow phenomena: (1) highly three-dimensional flow fields, (2) shock - boundary-layer interactions, and (3) shear layer control. Specific examples of current and future elements of the research program are described for each of these phenomena. In particular, the highly three-dimensional flow field phenomena is highlighted by describing the computational and experimental research program in transition ducts having a round-to-rectangular area variation. In the case of shock - boundary-layer interactions, the specific details of research for normal shock - boundary-layer interactions are described. For shear layer control research in vortex generators and the use of aerodynamic excitation for enhancement of the jet mixing process are described.

Future research in inlets, ducts, and nozzles will include more emphasis on three-dimensional full Navier-Stokes methods and corresponding experiments designed to concentrate on the appropriate three-dimensional fluid flow physics.

INLETS, DUCTS, AND NOZZLES RESEARCH PROGRAM

The internal fluid mechanics research program in inlets, ducts, and nozzles is described according to three types of fluid flow phenomena - highly three-dimensional flow fields, shock - boundary-layer interactions, and shear layer control. The importance of each of these flow phenomena is a result of the drivers listed on the figure. For example, highly three-dimensional internal flow fields result from unconventional engine locations where twisting and turning inlets, ducts, and nozzles must be designed to deliver the airflow to and from the free stream. Aircraft thrust vectoring requirements quite often lead to the transitioning of nozzle cross-sectional geometries from round to rectangular with resultant three-dimensional flows. Aircraft maneuverability requirements can lead to significant three-dimensional flow fields entering the propulsion system inlet and ducting system. The push toward higher flight speeds, for both military and civilian applications, leads to the importance of research in shock - boundary-layer interactions within inlets, ducts, and nozzles. The desire to design inlets, ducts, and nozzles to be as short and light as possible points to the importance of shear layer control as a means for "stretching" the limits of the geometry while avoiding internal flow separations. Shear layer control in another sense, that is, the use of aerodynamic excitation to control the formation and development of a mixing layer, offers the potential for enhancing the mixing process in external nozzle flows.

INLETS, DUCTS, AND NOZZLES

FLOW PHENOMENA	DRIVER
HIGHLY 3D FLOW FIELDS	UNCONVENTIONAL ENGINE LOCATION THRUST VECTORING MANEUVERABILITY
SHOCK-BL INTERACTIONS	HIGHER FLIGHT SPEEDS
SHEAR LAYER CONTROL	SHORTER/LIGHTER ENHANCED MIXING

CD-87-29854

SPECIFIC RESEARCH ELEMENTS FOR EACH OF THE THREE FLOW PHENOMENA

Specific elements of the inlet, duct, and nozzle research program are listed for each of the three flow field phenomena. The four elements highlighted in the figure will be expanded upon in the remainder of the presentation. Specifically, highly three-dimensional flow fields will be illustrated by describing the transition duct research program. The shock - boundary-layer interaction phenomena will be illustrated with a description of the normal shock - boundary-layer interaction research program. Shear layer control research will be illustrated with two examples of current programs: vortex generator research and enhanced jet mixing research. The remaining four elements of the overall program, that is, offset ducts, oblique shock - boundary-layer interactions, glancing sidewall shock - boundary-layer interactions, and boundary-layer bleed, will not be described in this presentation, although they are also key elements of the overall program.

INLETS, DUCTS, AND NOZZLES

FLOW PHENOMENA

- **HIGHLY 3D FLOW FIELDS**
  - TRANSITION DUCTS
  - OFFSET DUCTS
- **SHOCK-BOUNDARY-LAYER INTERACTIONS**
  - NORMAL SHOCK-BOUNDARY LAYER
  - OBLIQUE SHOCK-BOUNDARY LAYER
  - GLANCING SIDEWALL SHOCK-BOUNDARY LAYER
- **SHEAR LAYER CONTROL**
  - VORTEX GENERATORS
  - BOUNDARY-LAYER BLEED
  - ENHANCED JET MIXING

CD-87-29855

APPROACH TO RESEARCH PROGRAM IN INLETS, DUCTS, AND NOZZLES

The approach to the research program in inlets, ducts, and nozzles consists of a balance between computational and experimental research. Computationally, much of the emphasis until recently has been on the development and validation of parabolized Navier-Stokes methods. More recently and for the future, more emphasis is being placed on three-dimensional full Navier-Stokes methods.

Shown on the right-hand side of the figure are the elements of the experimental program in inlets, ducts, and nozzles. They are aligned with the respective computational method and point out the close linkage between the computational and experimental elements of the program. Note that each of the program elements listed in the previous figure under the three different flow phenomena also appear here as specific experiments.

INLETS, DUCTS, AND NOZZLES

APPROACH

COMPUTATIONAL METHODS	EXPERIMENTS
3D PARABOLIZED NAVIER-STOKES (PNS)	(1) TRANSITION DUCTS (2) OFFSET DUCTS (3) SHOCK-BOUNDARY-LAYER INTERACTION (4) VORTEX GENERATORS (5) BOUNDARY-LAYER BLEED
3D NAVIER-STOKES (NS)	(6) EXPERIMENTS (1) TO (5) (7) SEPARATION FLOW PHYSICS (8) NONORTHOGONAL SURFACES (9) JET MIXING

CD-87-29856

TRANSITION DUCT RESEARCH

Highly three-dimensional flow field research will be illustrated by describing the transition duct research program.

INLETS, DUCTS, AND NOZZLES
FLOW PHENOMENA

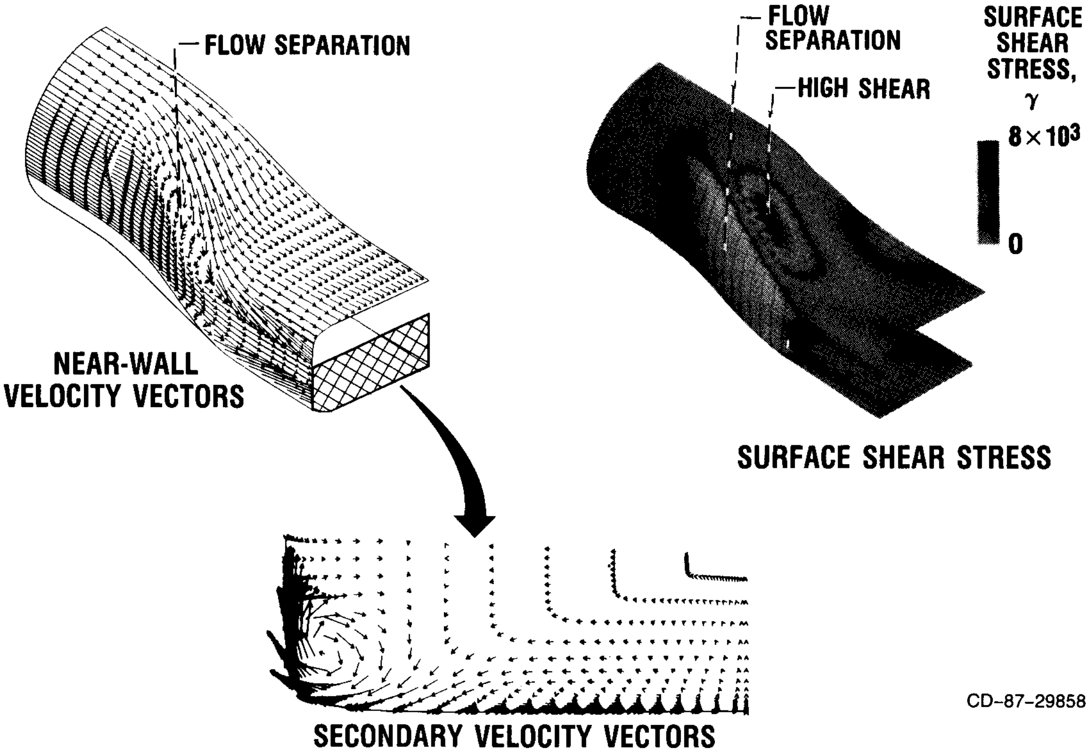
- **HIGHLY 3D FLOW FIELDS**
  - TRANSITION DUCTS
  - OFFSET DUCTS
- **SHOCK-BOUNDARY-LAYER INTERACTIONS**
  - NORMAL SHOCK-BOUNDARY LAYER
  - OBLIQUE SHOCK-BOUNDARY LAYER
  - GLANCING SIDEWALL SHOCK-BOUNDARY LAYER
- **SHEAR LAYER CONTROL**
  - VORTEX GENERATORS
  - BOUNDARY-LAYER BLEED
  - ENHANCED JET MIXING

TRANSITION DUCT, THREE-DIMENSIONAL, PARABOLIZED
NAVIER-STOKES COMPUTATIONAL RESULTS

A sample three-dimensional parabolized Navier-Stokes (PNS) computation for a specific transition duct geometry is illustrated in this figure. The geometry is described by a rectangular exit having an aspect ratio (width/height) of 3.0. The duct is 3 inlet diameters long and has an exit-to-inlet area ratio of 1.0. The figure shows near-wall velocity vectors on the left with a display of the secondary velocity vectors in the rectangular exit plane shown below it. The three-dimensional character of the flow field is clear. On the right side of the figure are contours of surface shear stress on the duct. Zero shear stress corresponds to the onset of flow separation on the duct surface. Note that the computation indicates that this particular geometry has a small, localized separation zone about half way down the duct. The flow reattaches just downstream of this zone and remains attached for the remainder of the duct length. The separation is apparent in both the near-wall velocity vectors and in the surface shear stress contours.

TRANSITION DUCT, 3D-PNS COMPUTATIONAL RESULTS

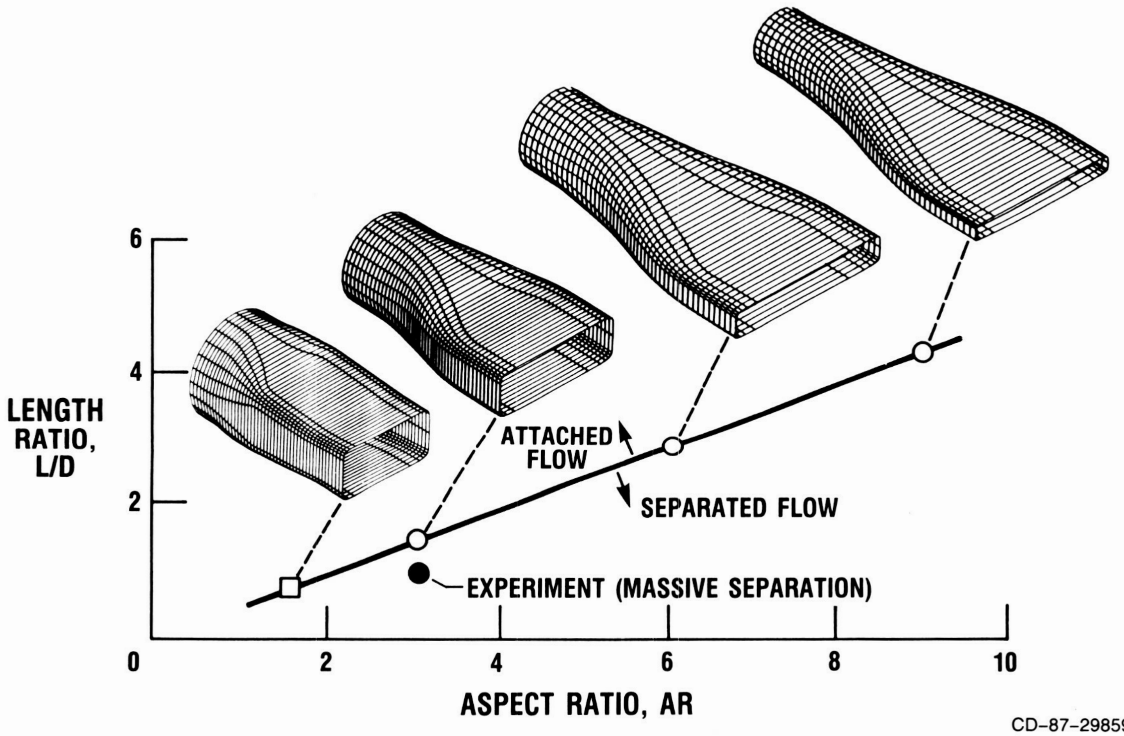
AR=3.0, L/D=1.5, A_{exit}/A_{inlet}=1.0



As an illustration of how the computation results shown in the previous figure can be used, this figure shows the internal flow separation bounds for a class of transition duct geometries. Each duct geometry has the same area ratio (1.0) but the length ratio and the exit aspect ratio are varied. Specifically, for four different aspect ratios, the length ratio was decreased from a value where the internal flow was completely attached to a value where the flow just began to separate. This series of computations then resulted in the separation bound shown in the figure. For geometries above the curve, the flow is attached, for geometries below the curve, the flow is separated. One experiment data point is spotted on the figure as a case where the flow was massively separated within the duct.

TRANSITION DUCT SEPARATION CHARACTERISTICS

3D-PNS COMPUTATIONS; $A_{exit}/A_{inlet} = 1.0$

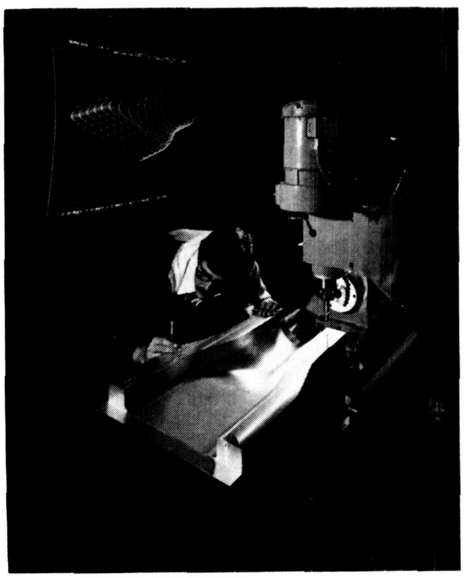


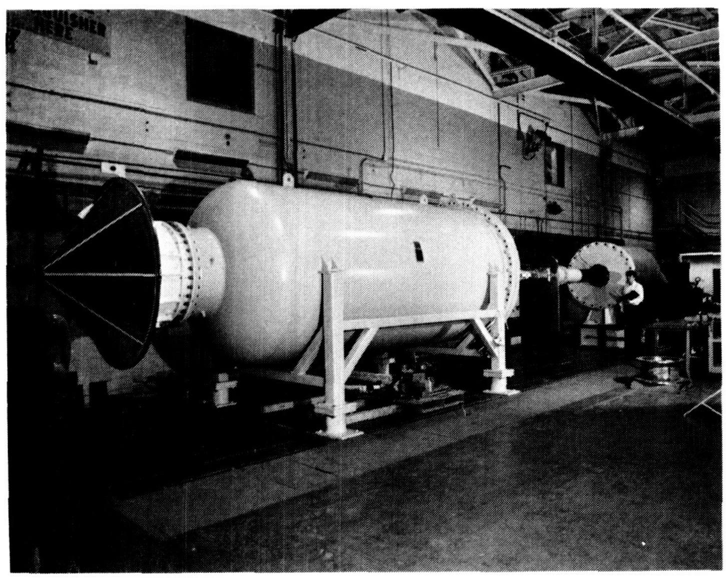
CD-87-29859

TRANSITION DUCT AERODYNAMIC EXPERIMENTS

To develop a more detailed understanding of the flow physics within transition ducts, to improve models of the flow physics, and to validate computations like those shown in the previous two figures, experiments are underway using the model and facility shown in this figure. The model was machined to match exactly one of the geometries for which the three-dimensional PNS method had been applied. Shown in this figure is a photograph of the model during the final machining stage along with a superimposed display of the computational surface. Also shown is the test facility. Special care was taken to condition the transition duct inflow properly to provide the desired levels of inflow turbulence, flow angularity, and boundary-layer profile. Modifications to the facility are currently underway whereby the inflow tank is being connected to a high-pressure supply system to provide higher Reynolds number test capability.

TRANSITION DUCT AERODYNAMIC EXPERIMENTS





CD-87-29860

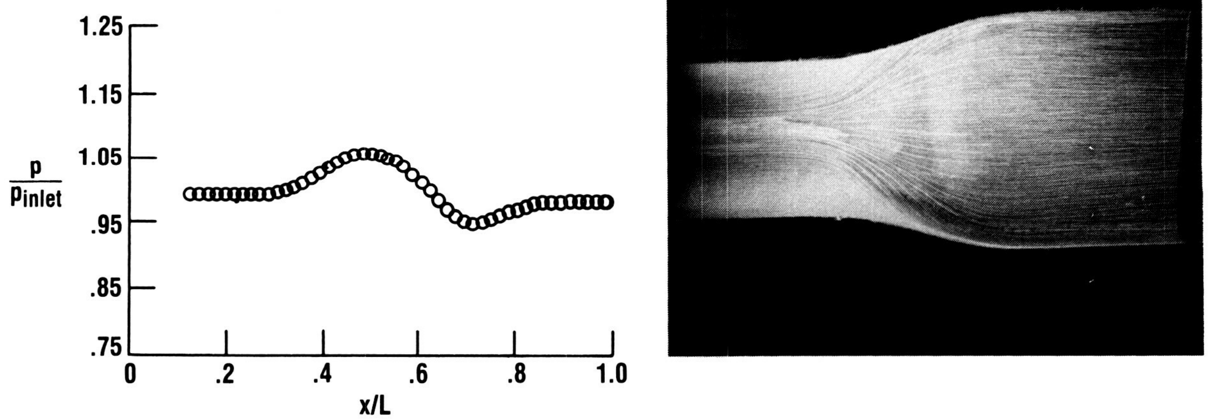
TRANSITION DUCT AERODYNAMIC RESULTS

Experimental results are shown in this figure at a one-dimensional duct Mach number of 0.5. On the left side of the figure, surface static-pressure measurements are shown along the centerline of the duct. On the right side of the figure are shown results from a surface-oil-streak flow-vizualization experiment. These oil streaks agree quite well qualitatively with the computational near wall velocity vectors. Although not shown here, experimental measurements have just been made of flow direction in the duct exit plane. Efforts are currently underway to compare those results with the corresponding three-dimensional PNS computations. Future experiments will include detailed probing of the flow field within this duct and others and direct measurement of surface shear stress using an advanced laser measurement technique.

TRANSITION DUCT AERODYNAMIC RESULTS

MACH=0.5

CENTERLINE SURFACE STATIC PRESSURE DISTRIBUTION | **SURFACE OIL STREAKS**

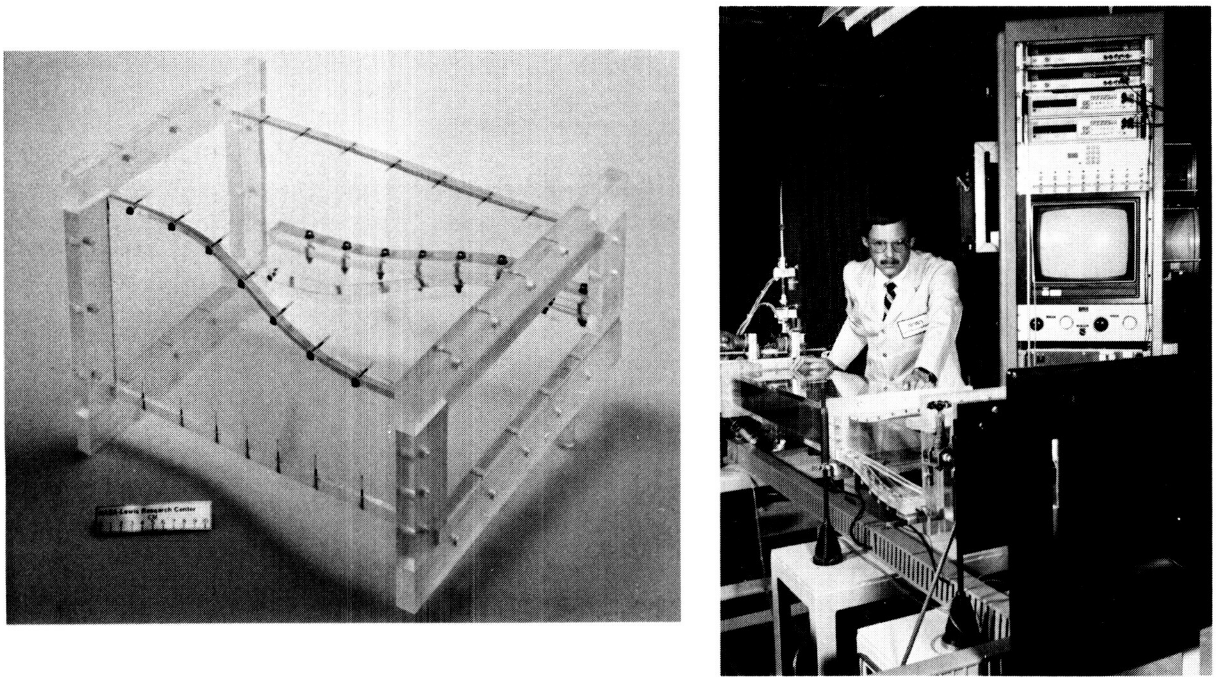


CD-87-29861

TRANSITION DUCT HEAT TRANSFER EXPERIMENTS

In addition to conducting aerodynamic experiments with the highly three-dimensional flow fields of transition ducts, heat-transfer experiments are also being conducted. Heat transfer is of particular concern for these types of flow fields because of applications where the three-dimensional flows may result in high-temperature flow streams (i.e., engine core flow) finding their way to the transition duct (nozzle) surfaces. The figure shows a square-to-rectangular transition duct model which was tested in the facility shown on the right. To measure the surface heat-transfer characteristics of the duct, the surface was coated with a liquid-crystal material. After establishing the desired amount of airflow through the duct, the duct surface was heated so that the resulting color bands (isotherms) on the liquid-crystal surface could be interpreted in terms of local heat-transfer coefficient.

TRANSITION DUCT HEAT TRANSFER EXPERIMENTS

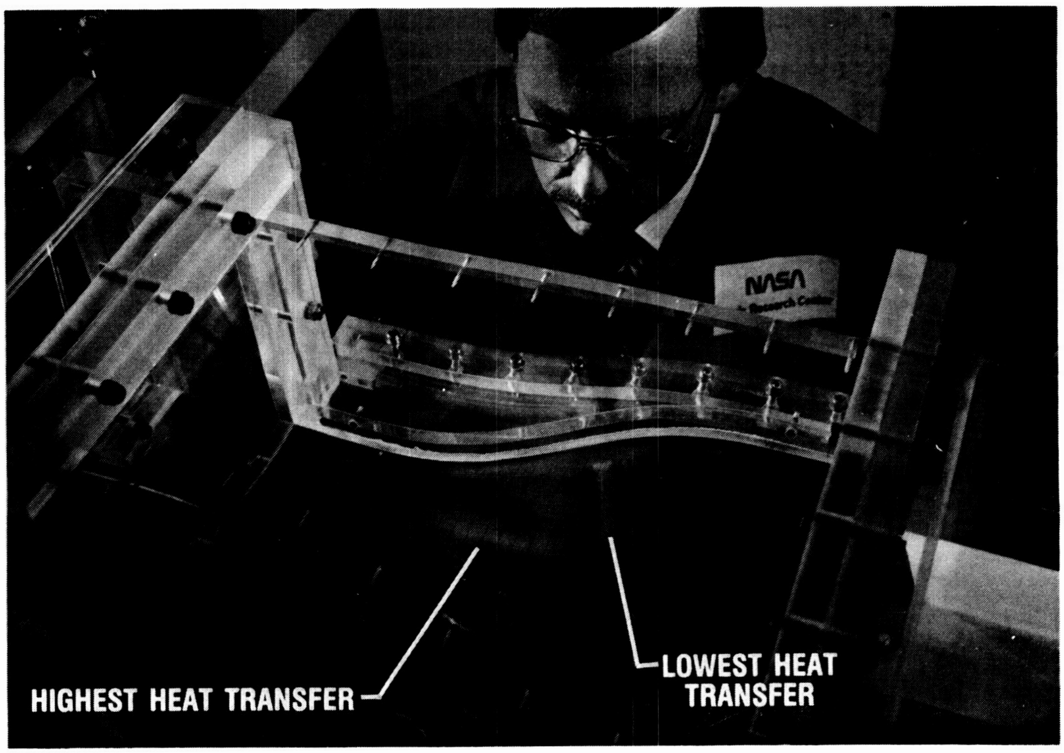


CD-87-29862

TRANSITION DUCT HEAT TRANSFER RESULTS

Results from the liquid-crystal heat-transfer experiment are illustrated in this figure. The different colors on the liquid-crystal surface correspond to different surface temperatures, which, when combined with the known level of surface heat input, lead to experimentally determined values of surface heat-transfer coefficient. Regions of high and low heat transfer are pointed out in the figure. By photographing the surface and then digitizing the photographic image, quantitative values of surface heat-transfer coefficient are obtained over the entire surface.

TRANSITION DUCT HEAT TRANSFER RESULTS



CD-87-29863

NORMAL SHOCK - BOUNDARY-LAYER RESEARCH

Shock - boundary-layer interaction research will be illustrated by describing the normal shock - boundary-layer interaction research program.

INLETS, DUCTS, AND NOZZLES

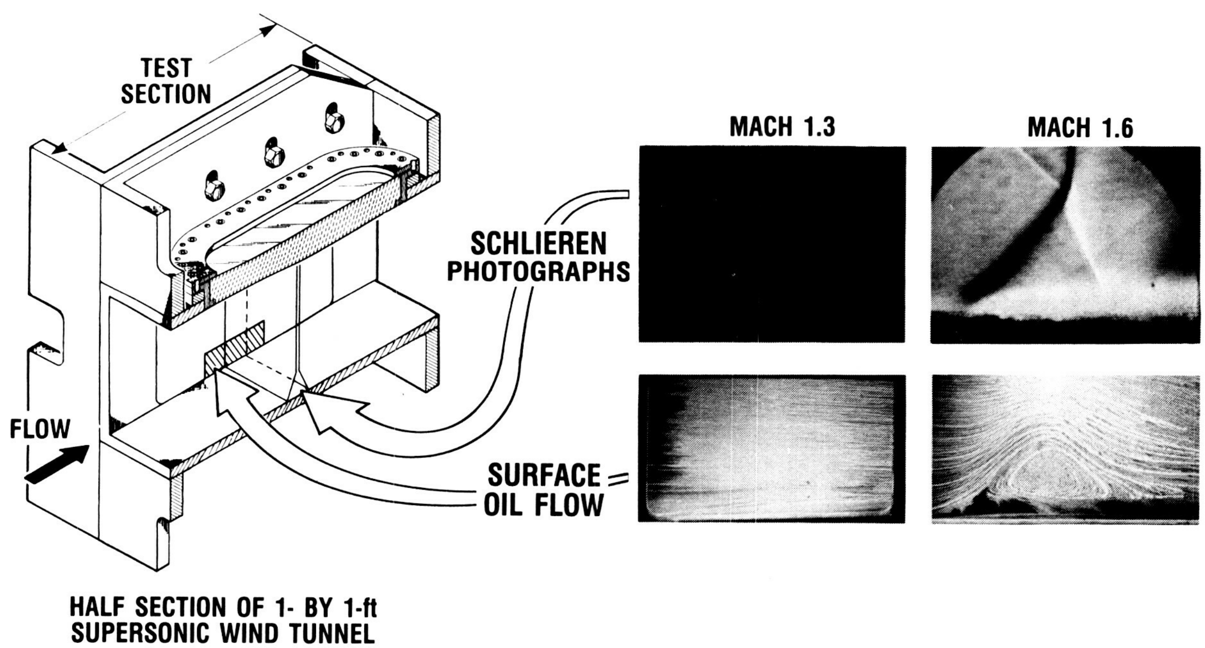
FLOW PHENOMENA

- **HIGHLY 3D FLOW FIELDS**
  - TRANSITION DUCTS
  - OFFSET DUCTS
- **SHOCK-BOUNDARY-LAYER INTERACTIONS**
  - NORMAL SHOCK-BOUNDARY LAYER
  - OBLIQUE SHOCK-BOUNDARY LAYER
  - GLANCING SIDEWALL SHOCK-BOUNDARY LAYER
- **SHEAR LAYER CONTROL**
  - VORTEX GENERATORS
  - BOUNDARY-LAYER BLEED
  - ENHANCED JET MIXING

NORMAL SHOCK - BOUNDARY-LAYER INTERACTION

This figure illustrates the nature of the normal shock - boundary-layer interaction. The interaction is shown at two Mach numbers, 1.3 and 1.6, and the nature of the interaction is illustrated with a schlieren photograph and a surface oil streak photograph at each Mach number. The photographs were taken in the Lewis 1- by 1-ft supersonic wind tunnel. The schlieren photographs give a view of the interaction integrated across the full width of the test section, while the surface oil streaks illustrate the details of the flow field along the side wall of the test section. At both Mach numbers the schlieren photographs show the shock being bifurcated into a "lambda shock" in the interacting region. The sidewall surface oil streaks indicate a significant difference in the structure of the flow field for the two cases. At Mach 1.3 the flow along the sidewall remains uniform and passes through the shock structure with no major alteration. At Mach 1.6 the adverse pressure gradient across the shock is strong enough to force the boundary layer to separate from the tunnel walls and to form the closed separation bubble shown in the figure. Note that this is a two-dimensional slice through a highly three-dimensional flow field that exists in the corner region.

NORMAL SHOCK–BOUNDARY-LAYER INTERACTION

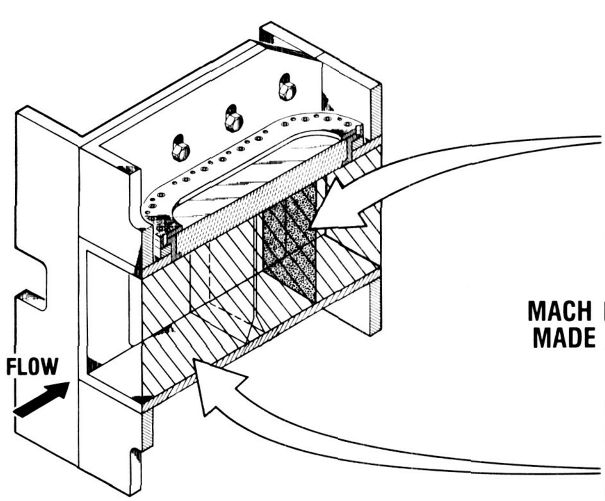


CD-87-29864

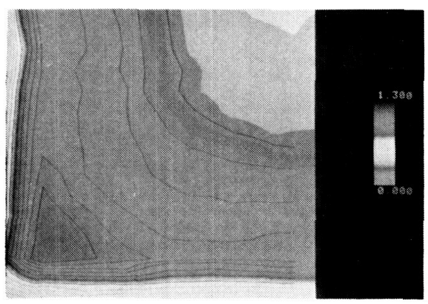
NORMAL SHOCK - BOUNDARY-LAYER INTERACTION -
LASER ANEMOMETRY RESULTS

Both the Mach 1.3 and 1.6 shock - boundary-layer interaction flow fields were surveyed in detail using nonintrusive laser anemometry. Results are shown for the Mach 1.6 case in two planes within the flow field. The top set of Mach contours illustrates the nature of the flow field in a cross plane downstream of the shock. The lower set of Mach contours shows the development of the flow field within a vertical plane passing through the centerline of the test section. The separated region in the vicinity of the initial interaction causes the actual flow area to contract downstream of the initial shock, leading to a reacceleration of the flow to supersonic Mach numbers. The flow then shocks down again, reaccelerates again due to the thickened boundary layer, and finally shocks down a final time.

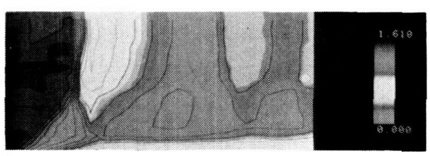
RESULTS OF LASER ANEMOMETER MEASUREMENTS FOR NORMAL SHOCK-BOUNDARY LAYER INTERACTION AT MACH NUMBER 1.6



HALF SECTION SHOWING TYPICAL MEASUREMENT PLANES



MACH NUMBER CONTOURS FROM MEASUREMENTS MADE ON CROSS-PLANE DOWNSTREAM OF SHOCK



MACH NUMBER CONTOURS FROM MEASUREMENTS MADE ON AXIAL MID-PLANE OF TUNNEL

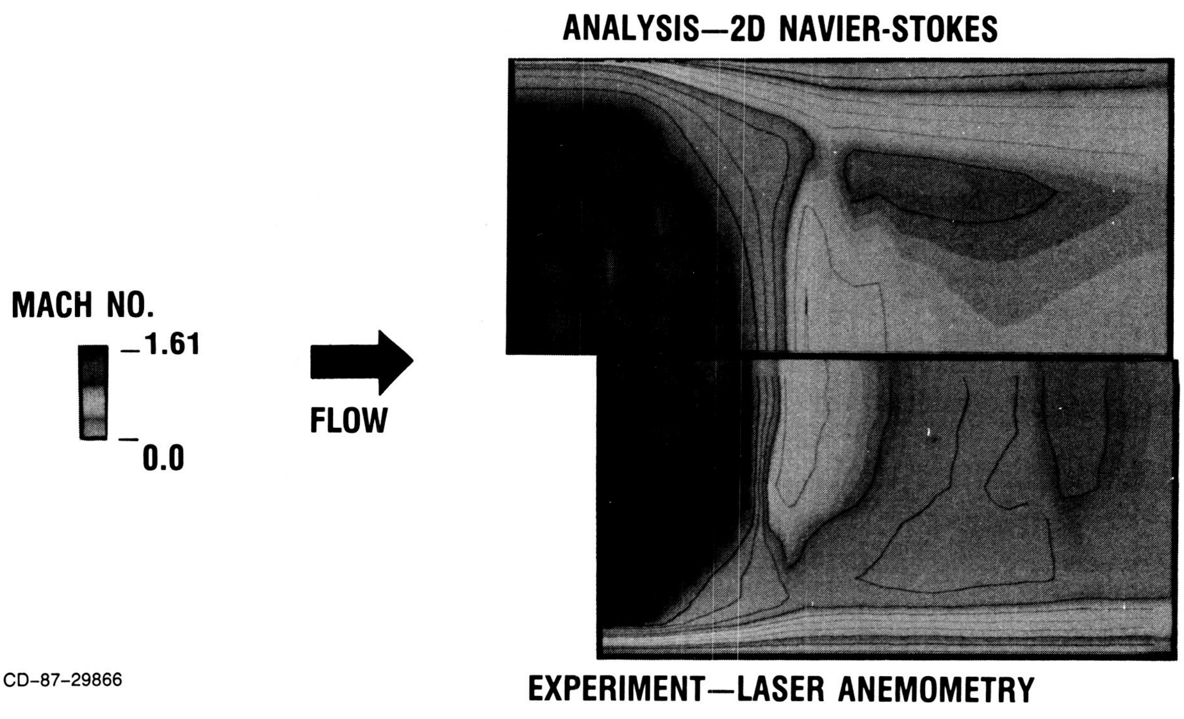
CD-87-29865

NORMAL SHOCK - BOUNDARY-LAYER INTERACTION -
COMPARISON OF ANALYSIS AND EXPERIMENT

The lower portion of the previous figure is repeated as the lower portion of this figure, that is, Mach contours measured with laser anemometry in a vertical plane passing through the centerline of the test section at Mach 1.6. The upper portion of the figure is a two-dimensional Navier-Stokes computation of the same flow field. Although the computation captures fairly well the initial portions of the flow in the vicinity of the first shock, in the downstream region, none of the flow physics are adequately represented. This result is, of course, expected since the experimental results have shown the strong three-dimensional character of the flow. This comparison points out the need for three-dimensional computational methods for computing such flow fields.

MACH 1.6 NORMAL SHOCK WAVE BOUNDARY LAYER INTERACTION

COMPARISON OF LDV EXPERIMENT AND ANALYSIS



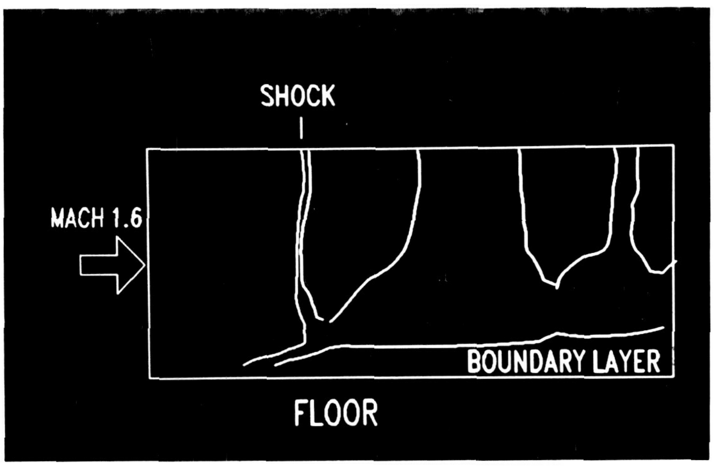
CD-87-29866

LASER ANEMOMETRY RESULTS PRESENTED WITH COMPUTATIONAL GRAPHICS

This figure is a frame from a film that will be shown during the conference. The film illustrates the additional insight and understanding that can be obtained from setting in to motion a three-dimensional contour representation of the flow field. The graphics software used to generate the film was actually developed for presenting computational results. The experimental laser anemometry data sets were modified to a format that was acceptable to the graphics software. The researcher can rotate the image, adjust the rate of rotation, and select the axis of rotation while sitting at the display console. One gains a perspective from these rotating images much more quickly than one would by looking at a series of two-dimensional or even three-dimensional plots.

NORMAL SHOCK-BOUNDARY-LAYER INTERACTION

LASER ANENOMETRY RESULTS PRESENTED WITH COMPUTATIONAL GRAPHICS PACKAGE



CD-87-29867

ORIGINAL PAGE IS
OF POOR QUALITY

VORTEX GENERATOR RESEARCH

The first aspect of shear-layer control, boundary-layer control, will be illustrated by describing the vortex generator research program.

INLETS, DUCTS, AND NOZZLES

FLOW PHENOMENA

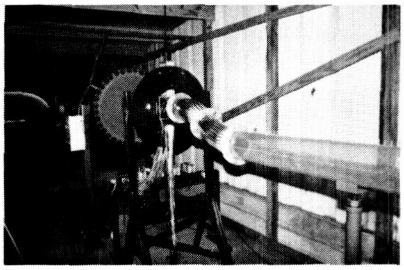
- HIGHLY 3D FLOW FIELDS
  - TRANSITION DUCTS
  - OFFSET DUCTS
- SHOCK-BOUNDARY-LAYER INTERACTIONS
  - NORMAL SHOCK-BOUNDARY LAYER
  - OBLIQUE SHOCK-BOUNDARY LAYER
  - GLANCING SIDEWALL SHOCK-BOUNDARY LAYER
- SHEAR LAYER CONTROL
  - VORTEX GENERATORS
  - BOUNDARY-LAYER BLEED
  - ENHANCED JET MIXING

VORTEX GENERATORS IN A DIFFUSING OFFSET DUCT

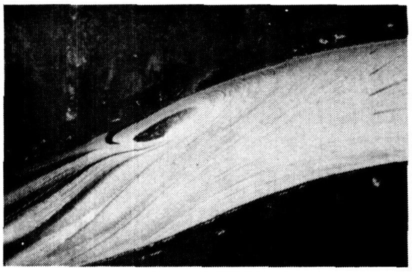
This figure illustrates a research model and facility that were used in an experiment to assess the performance of vortex generators in a diffusing offset duct. The duct had a length-to-diameter ratio of 5.0, an offset-to-diameter ratio of 1.34, and an exit-to-inlet area ratio of 1.50. Initial experiments with the duct identified the location of a separated flow region as shown by the surface oil streak photograph. Vortex generators were then added to the duct surface just upstream of the separated region to control the separation.

VORTEX GENERATORS IN DIFFUSING OFFSET DUCT

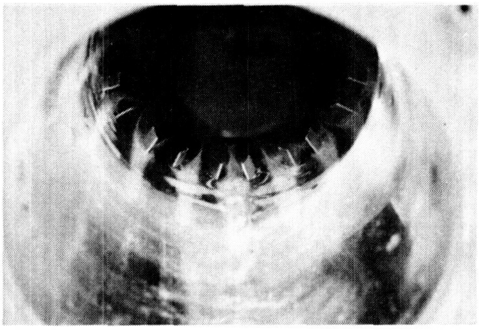
$L/D = 5.0$; $OFFSET/D = 1.34$; $A_{exit}/A_{inlet} = 1.50$



EXPERIMENT SETUP



SURFACE OIL STREAKS WITHOUT VORTEX GENERATORS



VORTEX GENERATORS

CD-87-29868

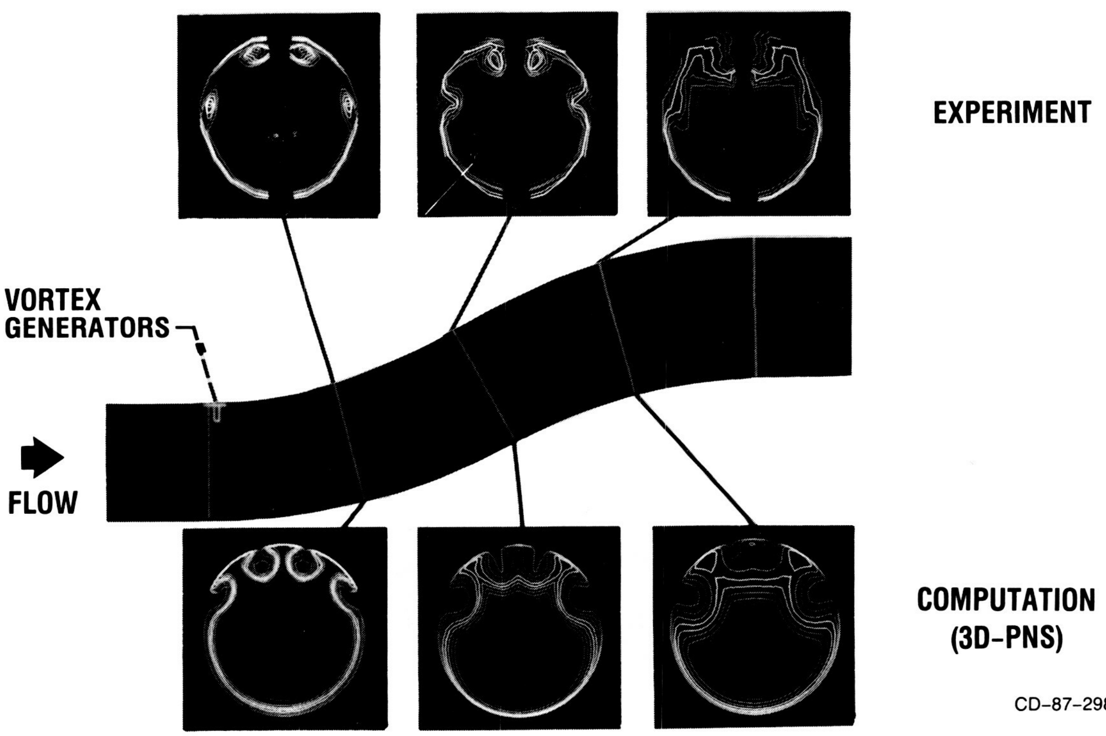
VORTEX GENERATORS IN OFFSET DUCT - COMPARISON OF ANALYSIS AND EXPERIMENT

A comparison of total-pressure contours at various locations down the length of the duct with the vortex generators in place, is shown in this figure. The experimental data were obtained by surveying the flow field with a total-pressure probe. The computational results are from the three-dimensional PNS method. The method includes a model for the vortex generators that allows one to position the individual generators anywhere within the duct and permits an adjustment to the strength of the vortex based on an empirical relationship with the vortex generator angle of attack.

In order to illustrate more clearly the influence of the vortex generators, a film will be shown of the total-pressure contours that one would see while moving downstream through the duct. Two segments of film will be shown - the first without the vortex generators and the second with the vortex generators.

VORTEX GENERATORS IN DIFFUSING OFFSET DUCT—
COMPARISON OF ANALYSIS AND EXPERIMENT

TOTAL PRESSURE CONTOURS; M=0.6



CD-87-29869

ENHANCED JET MIXING RESEARCH

Another aspect of the shear layer control, the use of aerodynamic excitation to control the development and evolution of large-scale structures in a shear layer, will be illustrated by describing the enhanced jet mixing research program.

INLETS, DUCTS, AND NOZZLES

FLOW PHENOMENA

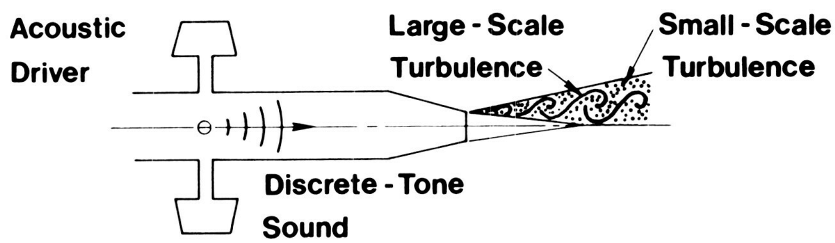
- HIGHLY 3D FLOW FIELDS
  - TRANSITION DUCTS
  - OFFSET DUCTS
- SHOCK-BOUNDARY-LAYER INTERACTIONS
  - NORMAL SHOCK-BOUNDARY LAYER
  - OBLIQUE SHOCK-BOUNDARY LAYER
  - GLANCING SIDEWALL SHOCK-BOUNDARY LAYER
- SHEAR LAYER CONTROL
  - VORTEX GENERATORS
  - BOUNDARY-LAYER BLEED
  - ENHANCED JET MIXING

ENHANCED JET MIXING

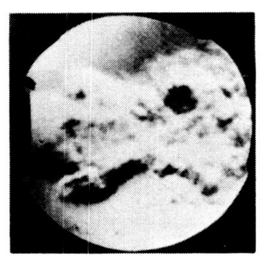
The mixing process between a jet and the surrounding air flow can be enhanced through use of aerodynamic excitation. The naturally occurring flow structure of the jet mixing process is shown in the left-hand Schlieren photograph. By applying an excitation signal at the proper frequency, here by use of acoustic drivers upstream of the jet exit plane, the naturally occurring large-scale structures within the mixing layer are regularized and enhanced and lead to a more rapid mixing process as illustrated in the center photograph.

ENHANCED JET MIXING

Coherent Large - Scale Turbulence in Jet Flows



Unexcited



Excited

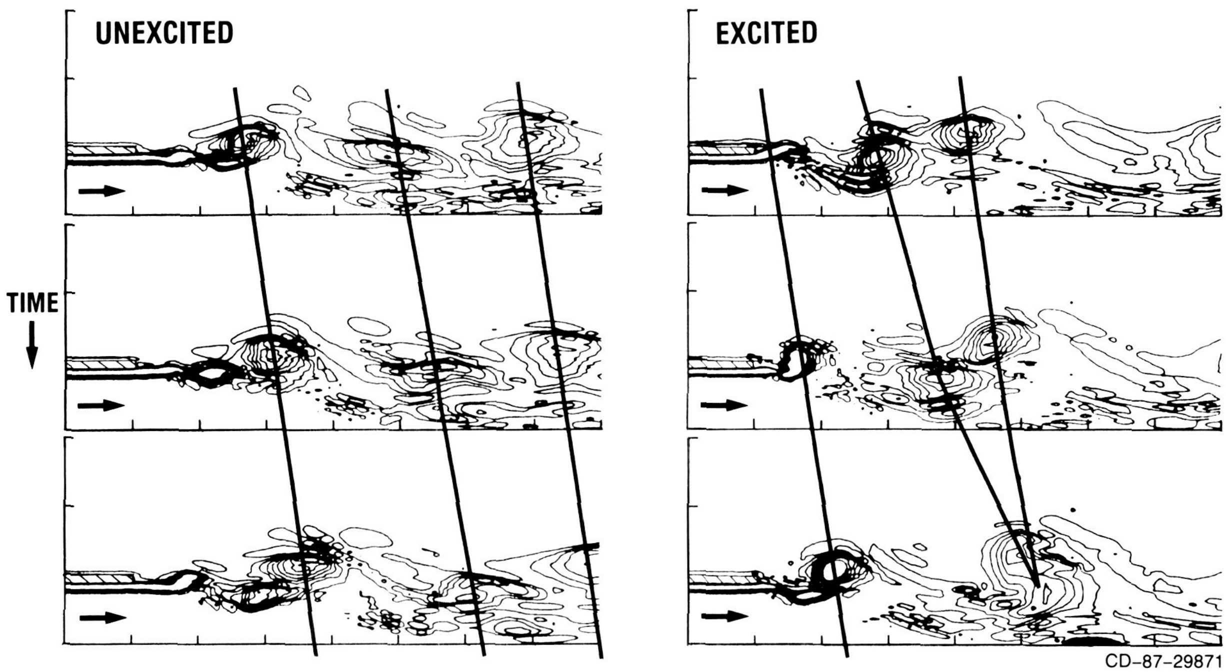
CD-87-29870

EFFECT OF EXCITATION ON JET SHEAR LAYER EVOLUTION -
AXISYMMETRIC NAVIER-STOKES COMPUTATION

Results of an axisymmetric Navier-Stokes computation of a jet flow exiting into a quiescent region are shown in this figure. The jet exit Mach number is 0.3, and the results are shown in terms of vorticity contours. Two cases are shown - on the left side an unexcited case and on the right side an excited case. For the excited case the excitation signal is applied at a frequency chosen to maximize the development and growth of the large-scale structures. In both cases results of the computation are shown at three different times. Also for both cases lines are drawn through the centers of the large-scale vortices to illustrate how they propagate downstream. Note that, with excitation applied at the proper frequency, adjacent vortices combine or pair to form a single-larger vortex, which, in turn, has the effect of more rapidly mixing the jet flow with the surrounding quiescent flow.

EFFECT OF EXCITATION ON JET SHEAR LAYER EVOLUTION— NAVIER-STOKES COMPUTATIONAL RESULTS

VORTICITY CONTOURS; MACH 0.3



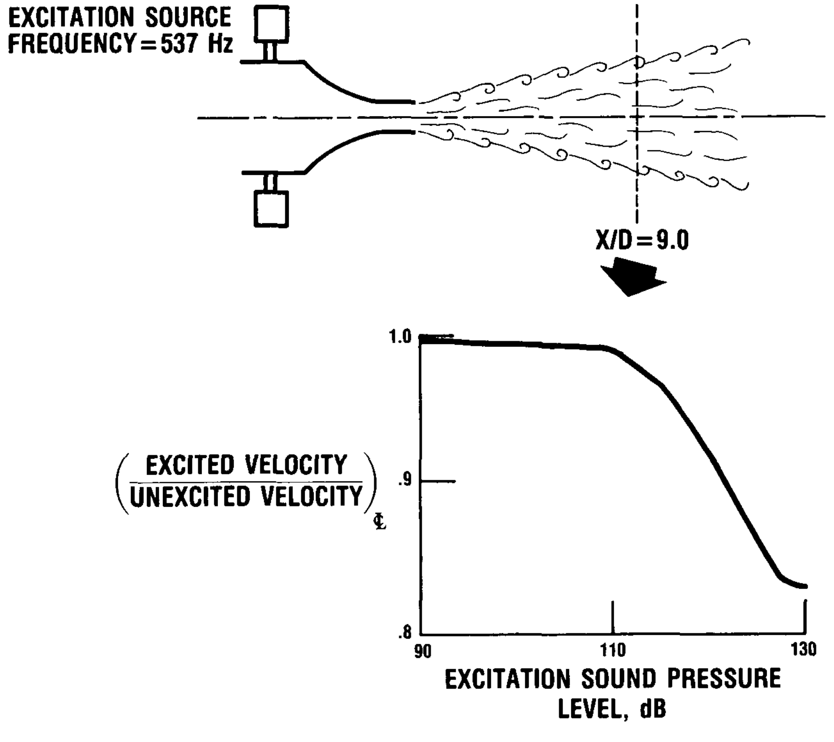
ORIGINAL PAGE IS
OF POOR QUALITY

EFFECTS OF EXCITATION ON JET MIXING - EXPERIMENTAL RESULTS

The effect of excitation on the jet mixing process for a jet having an exit Mach number of 0.3 and an initial turbulence level of 0.15 percent is shown in this figure. The results were obtained in an experiment wherein both the frequency and the level of the excitation signal could be varied. The results show how the ratio of centerline velocity for the excited case to that of the unexcited case varies as the level of the excitation is increased at a fixed frequency. The measurements are made nine jet diameters downstream of the nozzle exit plane. As in the previous figure, the excitation frequency has been selected to provide maximum mixing enhancement. As indicated, the effect of the excitation is quite significant with, in this case, a reduction in centerline velocity of about 16 percent at the maximum available excitation level of 130 dB.

EFFECT OF EXCITATION ON JET MIXING—EXPERIMENTAL RESULTS

MACH=0.3; JET TURBULENCE=0.15%



CD-87-29872

In summary, the internal fluid mechanics research program in inlets, ducts, and nozzles is a balanced effort between the development of computational tools and the conduct of experimental research. The program has been briefly described by highlighting research efforts in highly three-dimensional flow fields, shock - boundary-layer interactions, and shear-layer control. Much of the computational focus until now has been on the development and validation of parabolized Navier-Stokes methods. More recently and in the future, more emphasis will be placed on the development and validation of three-dimensional Navier-Stokes methods. The experimental element of the program will continue to provide a fundamental understanding of the fluid flow physics, to develop new and/or improved flow models, and to provide benchmark datasets for computational method validation of both the parabolized and full Navier-Stokes methods.

INLETS, DUCTS, AND NOZZLES
SUMMARY

COMPUTATIONAL METHODS	EXPERIMENTS
3D PARABOLIZED NAVIER-STOKES (PNS)	(1) TRANSITION DUCTS (2) OFFSET DUCTS (3) SHOCK-BOUNDARY-LAYER INTERACTION (4) VORTEX GENERATORS (5) BOUNDARY-LAYER BLEED
3D NAVIER-STOKES (NS)	(6) EXPERIMENTS (1) TO (5) (7) SEPARATION FLOW PHYSICS (8) NONORTHOGONAL SURFACES (9) JET MIXING

CD-87-29873